International Journal of Civil Engineering and Technology (IJCIET)
Volume 8, Issue 3, March 2017, pp. 1103–1111 Article ID: IJCIET_08_03_113
Available online at http://www.iaeme.com/IJCIET/issues.asp?JType=IJCIET&VType=8&IType=3
ISSN Print: 0976-6308 and ISSN Online: 0976-6316

MODELLING AND ANALYSIS OF REINFORCED CONCRETE BEAM UNDER FLEXURE USING ANSYS

Diyyala Naga Moulika
B. Tech Student, Department of Civil Engineering, KL University,
Vaddeswaram, Andhra Pradesh, India

Reshma Vasireddy
B. Tech Student, Department of Civil Engineering, KL University,
Vaddeswaram, Andhra Pradesh, India

P. Polu Raju
Associate Professor, Department of Civil Engineering, KL University,
Vaddeswaram, Andhra Pradesh, India

ABSTRACT

Understanding the behaviour of structural components like beam, column and wall during loading is crucial for the development of efficient and safe structures. In this article, the reinforced concrete beam has been modelled and analysed when subjected to two point loads at one third span from each support, using Finite Element Analysis tool, popularly called ANSYS software. The modelled and analysed beam having size 600 mm × 160 mm × 160 mm with 3 numbers of 12 mm diameter bars as main reinforcement, 2 numbers of 8 mm diameter as hanger bars and 8 mm diameter at 100 mm c/c as shear reinforcement. The behaviour of the analysed beam has been observed in terms of the flexural behaviour, crack pattern and displacement for various loading conditions such as 50 kN, 150kN, 250kN, 350kN, 450kN and failure load (690kN).Based on the analysis carried out on the RC beams using ANSYS, it is found that results are more sensitive to mesh size, materials properties, load increments, etc.

Key words: RC Beam, Finite element analysis, ANSYS, Flexural behaviour.

Cite this Article: Diyyala Naga Moulika, Reshma Vasireddy and P. Polu Raju, Modelling and Analysis of Reinforced Concrete Beam Under Flexure Using Ansys. *International Journal of Civil Engineering and Technology*, 8(3), 2017, pp. 1103–1111.
http://www.iaeme.com/IJCIET/issues.asp?JType=IJCIET&VType=8&IType=3

1. INTRODUCTION

Experimental study on flexure behaviour of the beam gives the exact behaviour of the structure but is time consuming and expensive [2]. Ansys is one of the tools used to determine the flexural behaviour of the beam .Ansys works on finite element method. Finite element analysis is used for evaluation of the structures gives accurate and fast results compared to experimental study.Finite element method is a numerical analysis method that divides the element into smaller parts and analyse the element under given loading conditions and hence evaluates the response of the material. The response of the element is represented in terms of finite number of degrees of freedom as the value of unknown function in set of nodal points. Most of the problems are non-linear in nature. Hence the non-linear analysis is an effective tool to obtain the exact solution. [2]. Non-linear analysis is a method that stimulates the exact behaviour of the material in inelastic range and to identify the potential of high load carrying capacity of the components through redistribution and shear strength. Finding of reinforcement in concrete, properties of concrete, meshing, incorporation of boundary conditions for the supports and modelling of loading and support regions, effect of shear reinforcement on flexure behaviour and other parameters which governs the analysis are considered for the present study [1].This paper deals with the nonlinear finite analysis using Newton-Rapson method to solve higher order differential equations [3].

2. DESCRIPTION OF THE BEAM MODEL

Experimental analysis is widely carried out to study individual component members under two point load conditions at one third of span. This method provides the actual behaviour of the structure, but it is time consuming and expensive. The development in the software modelling and analysis of structural elements has become simple. The finite element analysis is one such tool.

2.1. Geometry of the Beam

Dimensions of the RC beam consider for this study is (600mm x160mm x160 mm).The beam is simply supported with hinge at one end and roller at other end. Two point loads are applied at one-third of the span. The details of the beam are shown in Fig.1.

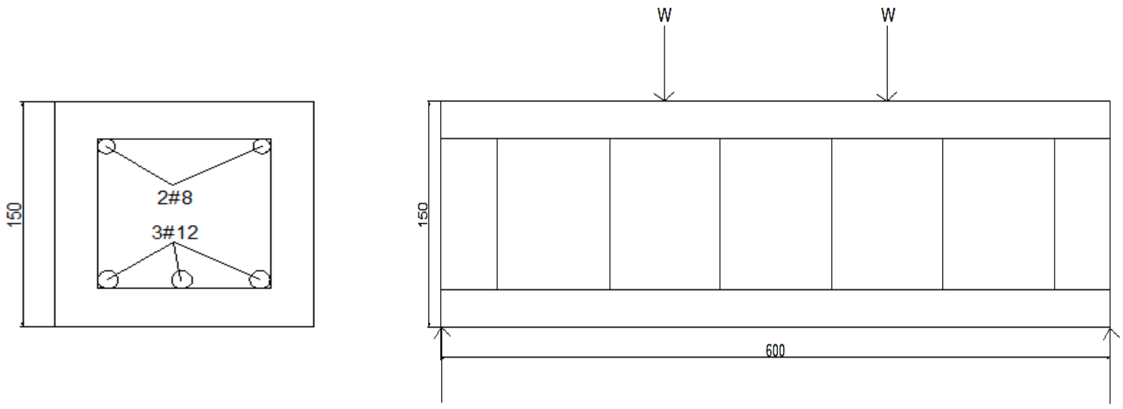

Figure 1 Beam Details

2.2. Material Properties

The grade of reinforcement bar used beam model was Fe415, Elastic modulus was 2×10^5 MPa and Poison ratio -0.3, whereas concrete grade was M25, Elastic modulus was 25000 MPa and Poison’s ratio was 0.15.

2.3. Element Types

The SOLID 65 is used for 3D modelling of concrete beams with or without rebars. The concrete is capable of cracking in tension and crushing in compression. In ANSYS SOLID 65 is used to model the concrete and LINK 180 is used to model the steel reinforcement. The geometry and node locations for this element are shown in Fig.2.The element is defined by eight nodes with three degrees of freedom at each node in the nodal x, y, and z directions [4]. The default element coordinate system is along global directions. The special feature in SOLID65 is cracking in three orthogonal directions, plastic deformation and crushing. The element types for this model are shown in Table 1.

Table 1 Element types for working model

Material type	ANSYS
Concrete	Solid65
Steel Reinforcement	Link180

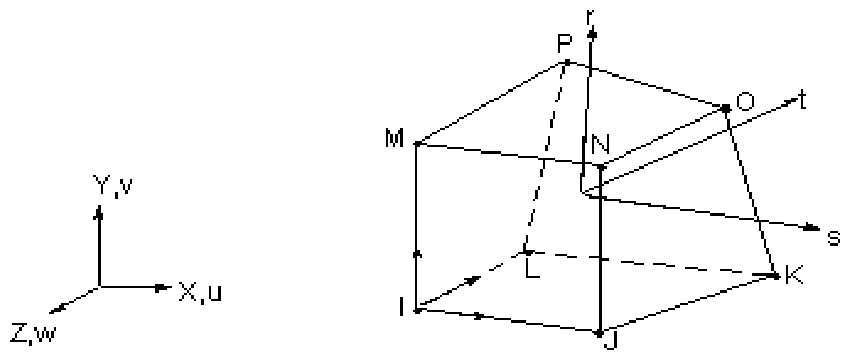

Figure 2 Solid65 Element

LINK180 is a 3D bar that is useful in a variety of engineering applications. The element can be used to model trusses, sagging cables, links, springs, and so on. The element is a uniaxial tension-compression element with three degrees of freedom at each node translations in the nodal x, y, and z directions [6]. As it is a pin jointed element, no bending of the element is considered. The real constants for rebars are used in beam given in Table 2.

Table 2 Real Constants

S.No	Element type	Property	Real constants for rebar 1	Real constants for rebar 2	Real constants for rebar 3
1	Solid65	Material No.	0	0	0
2	Link180	Area(mm)	113	0	0
3	Link180	Area(mm)	50.28	0	0

2.4. Modelling

Building a finite element model requires a more ANSYS user's time than any other part of the analysis. Then, define the element, type's real constants and followed by model geometry. A beam with the dimensions shown in Fig.1 has been drawn in the present case. The ANSYS version 12 has been used to model the beam and the analysis part also has been conducted in ANSYS. An initial load of 50 kN has been adopted to see the effect of load and deflection

[5]. The model is 600 mm long with a cross section of 160 mm × 160 mm.The finite element model is shown in Fig.3. The dimensions for the beam model are presented in Table 3.

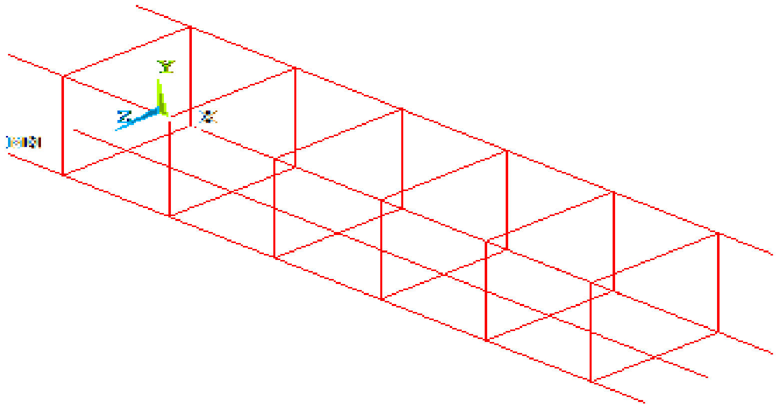

Figure 3 Reinforcement details of beam

Table 3 Dimensions for concrete

ANSYS	Concrete(mm)
X1,X2,X-coordinates	0,600
Y1,Y2,Y-coordinates	0,150
Z1,Z2,Z-coordinates	0,150

2.5. Meshing

In order to obtain the actual results from the Solid65 element, a mesh was recommended. The meshing of the reinforcement was a special case compared to volumes [6].The beam was meshed such that it is considered of square element of size 10mm. The necessary mesh attributes were set before the volume was meshed. The meshing of the beam is shown in Fig. 4. The merging of nodes and key points were carried out to avoid errors due to multiple nodes at the same location.

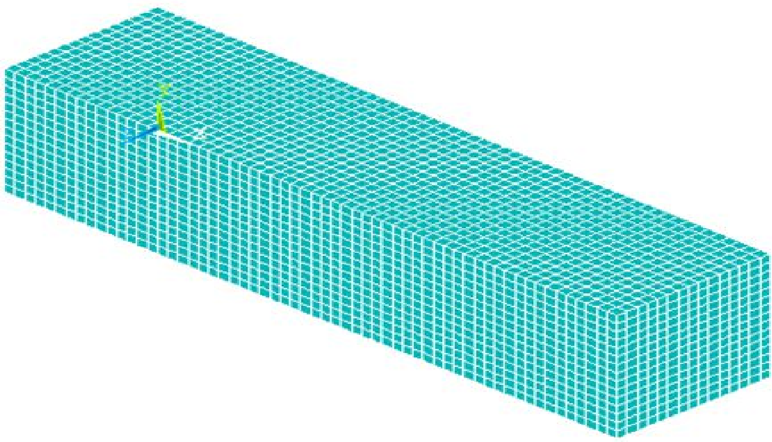

Fig.4 Discretization beam model

2.6. Loads and Boundary Conditions

Displacement conditions were needed to constraint the model to get a unique solution. The support was modelled as a fixed support at one end and hinged support at the other end. The external loads were applied as concentrated forces at equal one-third distance of the beam .The loading and boundary conditions of the beam were shown in Fig.5.

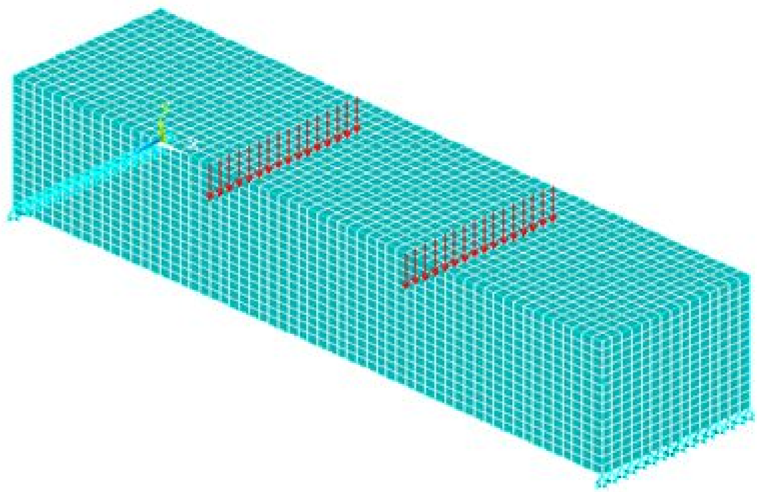

Figure 5 Finite Element Beam with Supports and Loads

For the analysis of the model, the static analysis type was utilized. The analysis was carried out for 'Small displacement static conditions. Then frequency is set to 'Write every Nth subset' and for different models maximum and minimum number of subsets were increased and the rest of the commands were set to the ANSYS default.

3. RESULTS AND DISCUSSION

While modelling RC beam for flexure analysis the beam is to be modelled including shear reinforcement using link 180 element so as to reflect the field reality [2]. Using the data obtained from the numerical analysis of the beam, displacement, stress intensity and the crack pattern in concrete are determined.Fig.6 indicated that at ultimate load level, there is a small variation in load vs displacement due to building up of more shear force. The stress intensity and crack pattern are shown in Fig.7 to Fig.12 and Fig.13 to Fig.18 respectively indicated that the load at first crack the for beams with shear reinforcement as marginally increased from 150 kN to 690 kN.

3.1. Variation of Displacement

Displacement is a degree to which a structural element is displaced under a load. It may refer to an angle or a distance. It is directly related to the slope of a deflected shape of the member under that load.The increase in the loading from 50 kN to 350 kN the displacement values increased and from 350 kN there is a decrease in the displacement value due to the increase in stress intensity and at the failure load the displacement value increased. The variation of displacement with respect to the load is shown in the below graph.

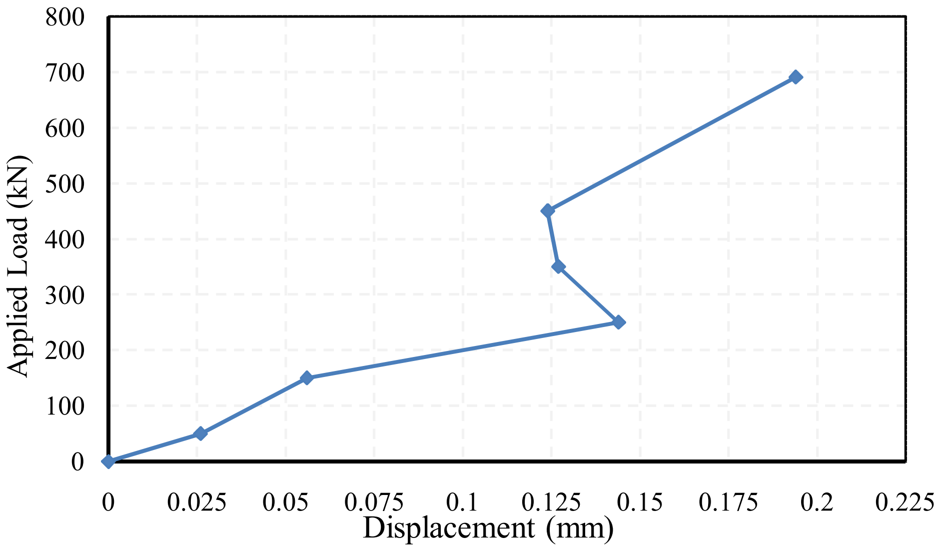

Figure 6 Applied Load vs. Displacement

3.2. Stress Distribution

Stress intensity is used to predict the stress state near the tip of a crack caused by a remote load or residual stresses. It is a theoretical construct usually applied to homogeneous, linear elastic material and it is useful for providing failure criteria for brittle materials. It can be applied to materials that exhibit small scale yielding at a crack tip.Fig.7 to Fig.12 shows the stress intensity near the supports and the load of simply supported beam for different loading conditions starting from 50kN to 690 kN. From the analysis of the beam under various loads it was found that by increasing the load, stress intensity is increased and is maximum at the bottom of the beam i.e. at the supports which is indicated by the red colour.

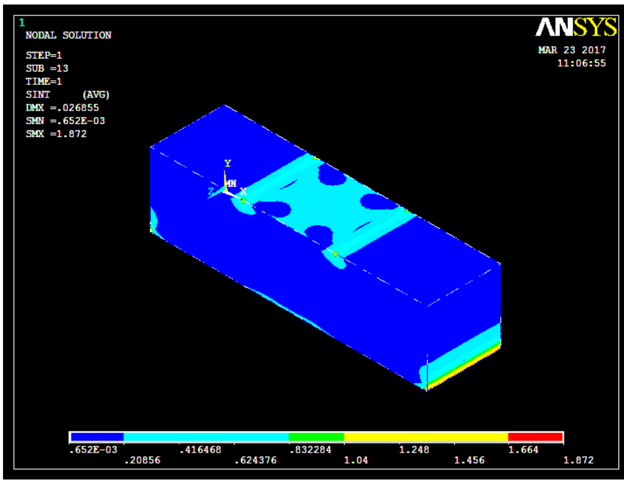

Figure 7 Stress Intensity of RC beam for 50 kN

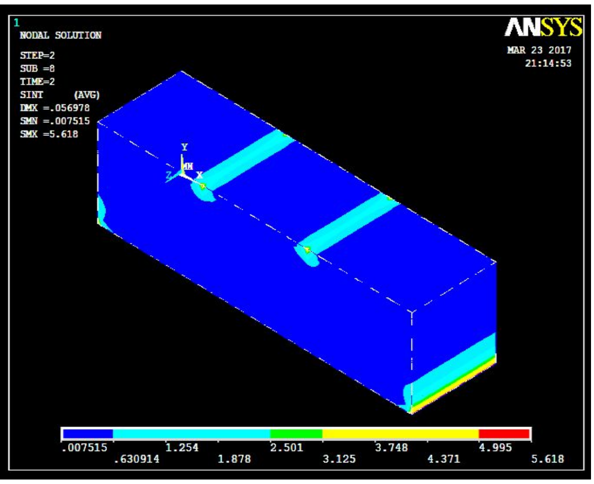

Figure 8 Stress Intensity of RC beam for150 kN

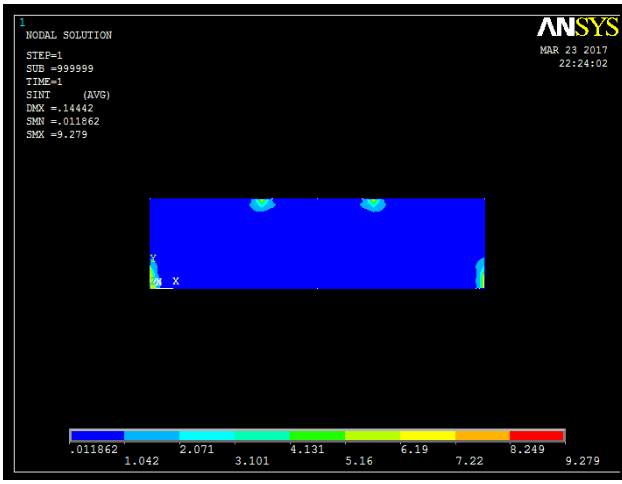

Figure 9 Stress Intensity of RC beam for 250 kN

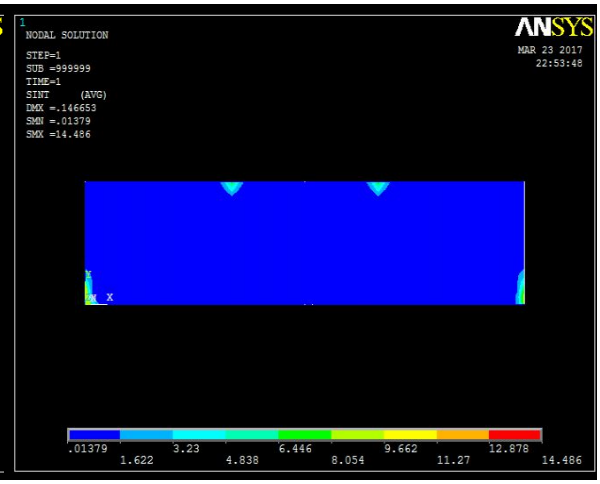

Figure10 Stress Intensity of RC beam for 350kN

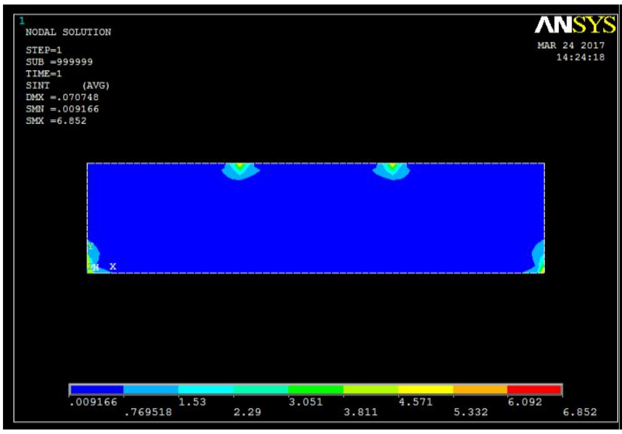

Figure 11 Stress Intensity of RC beam for 450 kN

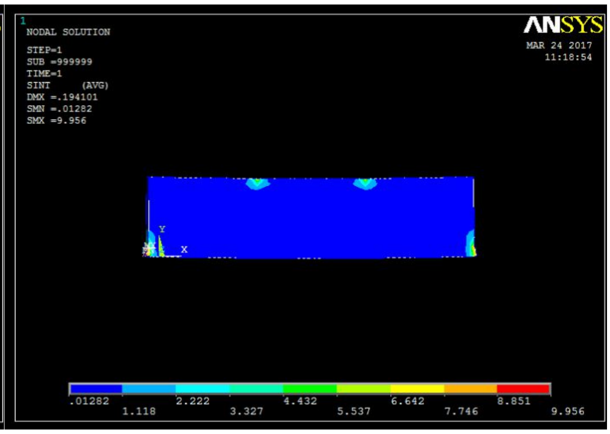

Figure 12 Stress Intensity of RC beam for 690kN

3.3. Crack Pattern

The model developed using ANSYS is capable of predicting failure for concrete materials. Both cracking and crushing failure modes are needed to define a failure surface for the concrete. Cracking occurs when the principal tensile stress in any direction lies outside the failure surface. Crushing occurs when all principal stresses are compressive and lie outside the failure surface. The following crack patters are observed under various loading conditions. In the non-linear region of the response subsequent cracking occurs as the more loads are applied to the beam. Cracking increases in the constant moment region, and the beam start cracking out towards the support at a load of 250 kN as in Fig.15. Significant flexural cracking occurred in the beam at more than 690 kN as in Fig.18 Due to the increase in the shear stress cracks are observed at the supports in Fig.15.

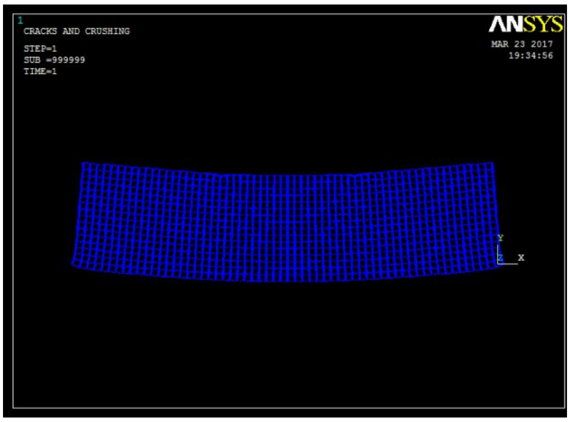

Figure 13 Crack pattern at 50 kN load

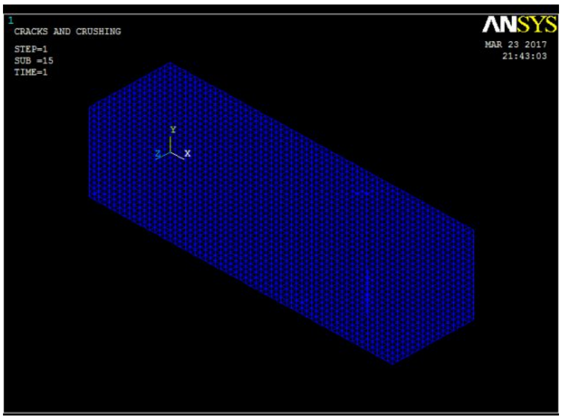

Figure 14 Crack pattern at 150 kN load

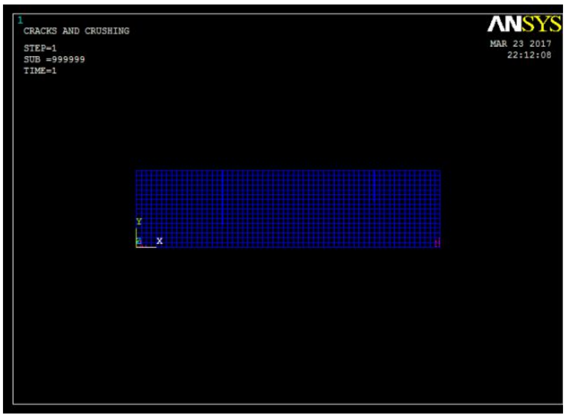

Figure 15 First crack at 250 kN load

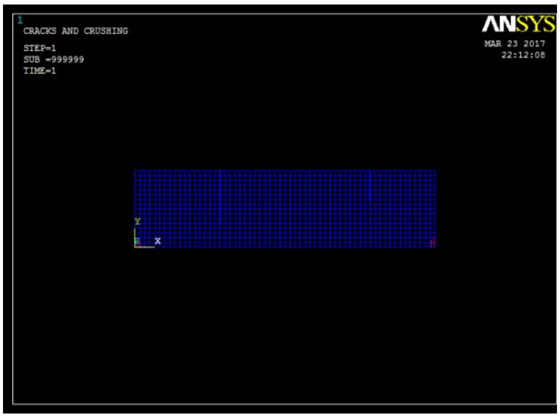

Figure 16 Cracking at 350kN load

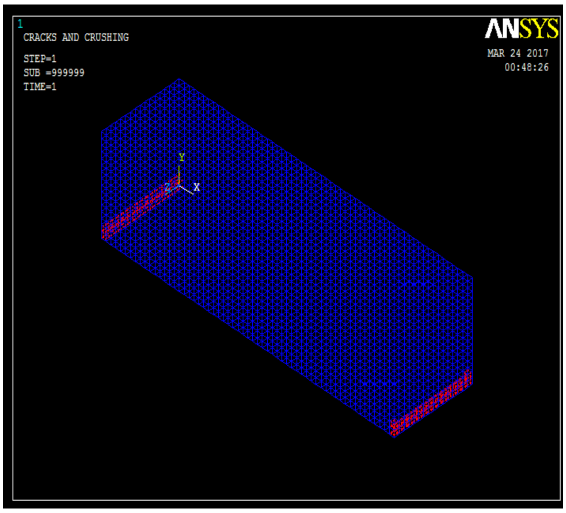

Figure 17 Cracking at 450 kN load

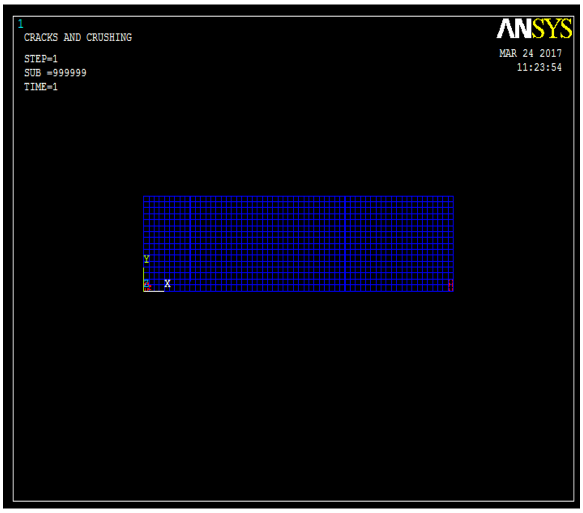

Figure 18 Cracking at 690 kN load

4. CONCLUSIONS

In this article, the behaviour of reinforced concrete beam is analyzed using finite element method. The parameters used in this study are magnitude of load. After compiling and analysing the model, the following conclusions may be drawn:

- Reinforced concrete beam can be modelled and analysed using ANSYS of version 12.0 software and obtain accurate results.
- From the analysis of the beam under various loads it was found that by increasing the load, stress intensity is increased gradually and is maximum at the bottom of the beam.
- From the analysis of beam under various loads it was found that with the increase in stress intensity the displacement value also increased.
- From the analysis of the beam under various loads it was found that by increasing the load, stress intensity is increased gradually and is maximum at the bottom of the beam.
- From the above results with the increase in the shear stress cracks are observed at the supports.

REFERENCES

[1] Vasudevan and Kothandaraman, G.(2011), "Parametric study on Nonlinear Finite Element Analysis on flexural behaviour of RC beams using ANSYS", International Journal of Civil and Structural Engineering,2(1),pp-98-111

[2] Neha, S. and Malipatil, M. (2014), "Parametric Study on Reinforced Concrete Beam using ANSYS", Civil and Environmental Research,6(8),pp-88-94

[3] Patil, S. and Niranjan, R. (2012), "Non Linear Finite Element Method of Analysis of Reinforced Concrete Deep Beam" International Journal of Modern Engineering Research,2 (6),pp-4622-4628.

[4] Subramanian, T. and Jayalakshmi, J. (2015),"Analytical Investigation of Bonded Glass Fibre Reinforced Polymer Sheets with Reinforced Concrete Beam Using Ansys", International Journal of Application or Innovation in Engineering & Management, 4(5),pp-105-112.

[5] Pradeep, S. Abhishek, M. and Arpit, K. (2016), "Finite element analysis of reinforced concrete beam using Ansys", International Journal of Current Engineering and Scientific Research,3(1),pp-120-126.

[6] Nimiya, R.J., Saibabu, S. and Siva, K.P. (2014), "Finite Element Analysis of Reinforced and Pre-Tensioned Concrete Beams", 4(10), pp-449-457.

[7] Prof. Dr. Ammar Yaser Ali and Ali Abdul Ameer Abdul Abbas, Structural Behavior of Reinforced Lightweight Concrete Corbels Strengthened with NSM-CFRP Bars. *International Journal of Civil Engineering and Technology (IJCIET),* 7(4), 2016, pp.380–392.

[8] N. Sundar, P. N. Raghunath and G. Dhinakaran, Flexural Behavior of RC beams with Hybrid FRP Strengthening. *International Journal of Civil Engineering and Technology*, 7(6), 2016, pp.427–433.